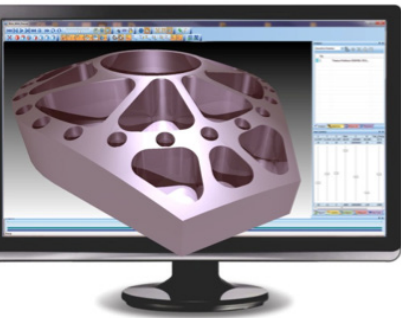




Improving CNC Programming Workflow
With CAD/CAM Software



Improving CNC Programming
Workflow With CAD/CAM Software



CAD/CAM Implementation

The implementation of CAD/CAM technology into your cnc machining workflow-process can make the difference in producing perfect parts efficiently while maintaining and improving the high level of quality that you need each step of the way. While the idea behind the development of CNC Software is purely automation oriented, shops around the world still struggle to become profitable, efficient in their cnc machining process while trying to maintain an edge competitively in their respective markets. Therefore, let's take a closer look at the subject of CAD/CAM software and how the implementation of it can contribute to improving programming workflow as it continues to prove an excellent solution for thousands of shops around the world in cnc manufacturing.

CAD Files & Compatibility

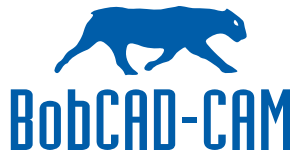
So how does CAD/CAM software improve production workflow? First we have to dissect each stage of making a perfectly finished part with a computer aided design and manufacturing software product. This process begins with how you acquire and manage your part model. With today's advancements in hardware design software technology, most cnc manufacturing businesses start with a 3D model of a part either designed by you in your CAD software from a print or provided by a client. SolidWorks, AutoCAD, Pro Engineer and SpaceClaim are examples of quality Design products that offer simple, yet highly advanced design and innovation solutions in the world today. These products are becoming more and more affordable and can even allow teams of designers to collaborate on assembly projects from different locations, solve geometry issues, repair bad or dirty geometry easily and just about anything else you can imagine within the scope of design and innovation. Therefore it's critical that a CAD/CAM system provide the ability to import or open CAD design files. These CAD design systems generally have their own native file types such as the SolidWorks SLDPRT type or AutoCAD DGN. To directly import the native file type would suggest that the likelihood of geometry problems would be slim to none is correct. The potential for issues can increase once a CAD file has been created and exported into another software product. Yet, CAD software products do allow users to save designed part files as a type of file that is acceptable to another product easily. While this is a complete subject in itself, a good quality solid model is a good quality solid model whether it is saved out as an STL file or a STEP file from a CAD system.

Among the most common are STL, STEP, IGES, DXF, DGN, SLDPRT and others. A CAD/CAM program not only offers import capabilities, they generally allow you to design as well. Because of this, a full library of drafting and solid or surface editing tools are made readily available in the software in the case that changes take place during production. Workflow improvements can be achieved through seamless file translation, part orientation in the viewing workspace, part file editing and geometry verification.

Using a CAD design product that offers CAM plug-ins will accelerate workflow efficiency such as SolidWorks and the BobCAM for SolidWorks Gold Partner CAM product. This means that designed part files do not have to be exported into another CAM product, reducing file, surface and geometry issues and streamlining the design to machining process.

CAM Job Management & Setting Up Your Stock

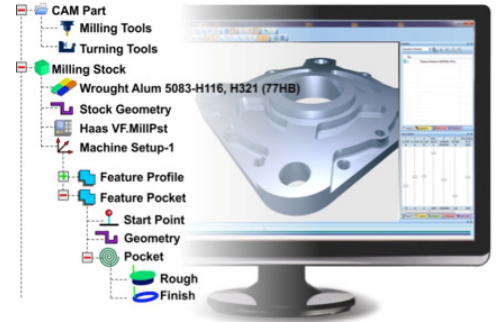
Once the part file is ready to move into the toolpath and programming stage you will find that most CAM (Computer Aided Manufacturing) software products provide a standard "Job Tree" method of organizing machining strategies into a productive workflow so that tools can be added, toolpath strategies can be used and a program can be created in its proper sequence.



Improving CNC Programming Workflow With CAD/CAM Software

The "Tree" organizes everything, allows you to turn machining operations on and off, re-order operations, post process and basically manage everything about the job program right within the software interface. So what would be the optimum order of production in this stage of the workflow process? The first things to be done would be the creation of the Stock for your part so that a Machine Setup - work coordinate can be created, then what your material and tools being used as this affects the speed and feed settings within cnc toolpath strategies. This is where a CAD/CAM system can improve efficiency in your workflow by adding a Stock Wizard that steps you through the process of creating your stock. This includes offering a full material library that can be customized and added to as needed.

Workflow improvements also take place in the ability to easily access and setup your Cutting Conditions and Tool Patterns (Multi-Tool operations such as a counter bore or hole drilling with a center drill, drill and possibly a chamfer), Tool Crib and Tool Holder data. All of this is important to creating a error free NC Program. Lastly, you will need to access a tool library for all of your tools. Rather than to talk about the types of tools we will just say, the more the better for all applications.

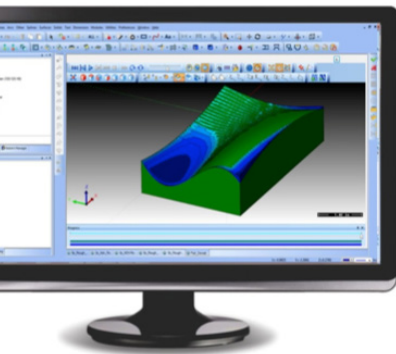


However the tool library should include all tool data, material, diameter, angle, length, label and more that you will be using for all machining strategies within the job tree manager. All of this information makes the process of setting up a job faster if the process is Wizard driven or if you have the ability to save set up details globally or separately from job to job. Another effective feature for saving time is the ability to save and load machine features. Many shops do a lot of the same types of jobs.

If you have already setup your CAM for one job and can use that setup for another similar job then you can just open the configuration and begin creating toolpath or at least not have to do so much setup. The idea here is "automation" so every opportunity to increase efficiency counts.

Part of creating and setting up your stock is the Machine Setup. The machine setup is part of the process and CAM software allows you to select an origin or create one by using X, Y and Z coordinates. You may also select an existing User Coordinate System (UCS) and also create a work offset for the program. The Machine Setup should be located within the Job Tree - Job Manager and give you the ability to edit it later, choose what machining operations you want to use (from 2, 3, 4 and 5 axis axis machining), add an index for 4th axis machining, load previously saved machining features and more. You should even be able to insert other Machine Setups with different Work Offsets. These software capabilities will positively affect the efficiency of your workflow when made available in an easy to understand and use interface.

Once you have set up the stock, tools, material and cutting conditions you want to set up the post processor for your job. The post processor is the translator used in the process of converting the setup data and toolpath data and creating a numeric language (G-Code) then you will send to the machine tools controller to physically machine the part. Most CNC Software systems offer post processors. Some offer the ability to customize a post processor so that it meets the requirements of the controller on the machine. Check with the software provider for a library of post configuration files, particularly the one for your specific machine tool.





Improving CNC Programming
Workflow With CAD/CAM Software



While this is one of the biggest topics of CAD/CAM software and why CAD/CAM even exists, Workflow efficiency can be achieved just by having the software provider assist you in the creation or modification of posts. A correct post processor eliminates the need for you to have to hand edit code. Thus, allowing you to spend your time focusing on the machining side of it all.

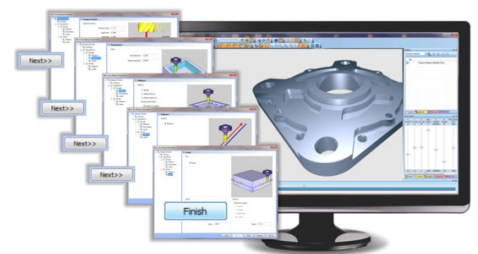
Toolpath & Machining Strategies

The next stage in the process of using CAD/CAM software involves the creation of Toolpath (cutter-path) which is the actual path that the cutting tool takes to machine the part. A CAM Tree-Job Manager will help you keep each machining operation organized and properly sequenced so that the g-code program produces the best possible cycle time for the job. 2, 3, 4 & 5 axis toolpaths are all developed to produce efficient roughing and finishing cycles. The Machine Setup should already be created in the Job Tree because you setup your stock. A Machining Wizard is a series of dialog boxes that step you through a process with a "Next" button until done where a "Finish" button would close the wizard and save everything you set for a toolpath machining strategy. When Wizards are used in the creation of toolpath two things happen.

New users create programs in a fraction of time over toolpath creation without using wizards and secondly, nothing is forgotten or left behind in making toolpath that is accurate and properly calculated based off of the tools you setup for the job. Toolpath Wizards start by allowing you to choose the areas of the part to apply a specified toolpath (planar, spiral, z-level etc) within a boundary or an entire part, set clearances and height of rapid moves for the tool.

Then you can choose the type of cutting pattern (zig-zag or single direction cutting), cutting direction (climb or conventional), roughing parameters, custom tolerance and cut-depth options, lead-in and out options, toolpath linking options (how you want to connect one path to another) and other custom machine options. When finished you will automatically generate the toolpath so you can visually inspect it. This takes place for each separate operation. If a change is needed you can simply go back into the operation and edit it. Then re-generate the toolpath. Once you have created toolpath for each operation you are ready to move into the next phase of the process.

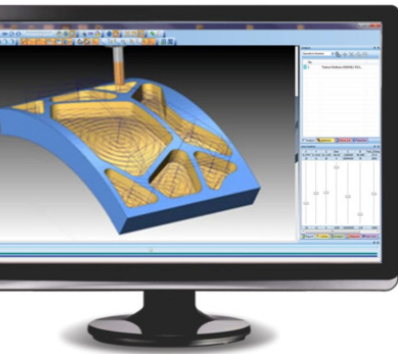
Improvements in workflow efficiency can also be accomplished by using High Speed toolpaths. High Speed machining is a revolutionary method of cutting where the traditional hard right and left turns in the cutting path are changed to produce more rounded or circular style cutting movements. In some cases the high speed cut paths are seen to be longer in overall length with this type of cutting path. However the benefits can be seen in that there is a constant tool engagement (less tool-material collision) which is proven to extend the life of cutting tools. In addition, cutting speeds are typically increased (High-Speed) which proves to reduce overall cycle times. High Speed toolpath was initially developed for 2 Axis machining (pocketing type operations).



As CAD/CAM providers continue to develop their technology you can now see high speed toolpaths available for more complex 3 Axis machining such as the advanced 2 & 3 Axis high speed Adaptive Roughing strategy that BobCAD-CAM software uses. This provider has recently released their Multiaxis CAD/CAM software that uses a High Speed Adaptive Roughing toolpath strategy that is the first in the world for Multiaxis CNC milling.



Improving CNC Programming
Workflow With CAD/CAM Software



Other toolpath strategies can include the Advanced Roughing option in BobCAD-CAM that allows the user to enter an Intermediate Step variable which tells the software to rough down to the bottom of a cavity and then come back up the outer walls stopping at each specified intermediate level and remove the material that tends to look like a large stair step on the walls of the part. This improves roughing quality and cuts down the amount of material that the finishing tool has to remove, all in one operation. This is another popular toolpath operation that will help increase efficiency. Lastly, workflow efficiency can be achieved in what is called, "Toolpath Associativity." Associativity is the ability to make a design change to a part and automatically update toolpath operations without having to re-create an entire job. This is highly advantageous when there are many operations within a single job.

Simulation & The G-Code Program

The final stage in the CAD/CAM programming phase is typically the toolpath simulation & creation of an NC Program file that can be sent to the machine tool. The simulation process allows the programmer to visually see the process and catch any errors within the program, gouges or collisions as an error detection stage. Simulation will also offer the ability to calculate cycle times as well as other benefits such as "machined part deviation" where you can visually inspect the part for areas that cutting tools did not fit in or cut during the machining process. Typically a number-color system is used to visually detect deviation areas, tools and machining operations.

Advanced Machine Simulation allows you to use your machines Kinematics to simulate the actual machines motion as well as the toolpath operations. The final stage is the generation of the NC File that is sent to the CNC Machine. Without a g-code program CAD/CAM is incomplete. This is a critical stage of the process due to potential loss of valuable time due to hand editing g-code programs. Once a completed post processor is created, a CAD/CAM product should allow you to automatically create code that is clean and edit free. RS 232 and DNC communications is then used to send the completed program to the cnc machine tool which is used to cut the part from the specified material and with the specified tools that you input.

Improving Efficiency & Beyond

These are the general stages of CAD/CAM cnc programming process along with examples of how you can improve the efficiency of your programming workflow by using it. While it is easy to say that CAD/CAM software should be easy to use, what is equally important is training availability. Does the software come with training. If it does, what forms is it available in. Are there training videos that allow you to learn on your own time? Are there local seminars or one-on-one web based training sessions available for you to learn on the fly and on your own time? The more options the better which includes the ability to achieve operator certification as a professional. We hope that this has enlightened you on the efficiency value of CAD/CAM Software for cnc machining.

For more information on implementing CAD/CAM software into your business call BobCAD-CAM, Inc. at 877-262-2231 or 727-442-3554. Visit www.bobcad.com for a free demo.

World Leader In Powerful & Affordable CAD/CAM Software

28200 U.S. Highway 19 N., Suite E

Clearwater, FL 33761



Toll Free: 877.262.2231

Intl: 727.442.3554

www.bobcad.com

